

FreeCAD-CFD Workbench

Tutorial 2: Multi-region mesh generation and porous media



CFD Workbench

WORKBENCH:

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

PREREQUISITES:

WINDOWS:

- Install the binary. All necessary software components are included.

LINUX:

- FreeCAD (https://www.freecadweb.org/wiki/Install_on_Unix)
- OpenFOAM (3.01+) (<https://openfoamwiki.net/index.php/Installation/Linux>)
- Paraview (tested with 5.0.1)
- Gnuplot (tested with 5.0)
- PyFoam (0.6.6 +)
- GMSH (2.13+)

For more information, view the CFD workbench README file.

DEVELOPERS:

Johan Heyns (CSIR, 2016) jheyns@csir.co.za,

Oliver Oxtoby (CSIR, 2016) ooxtoby@csir.co.za,

Alfred Bogaers (CSIR, 2016) abogaers@csir.co.za,

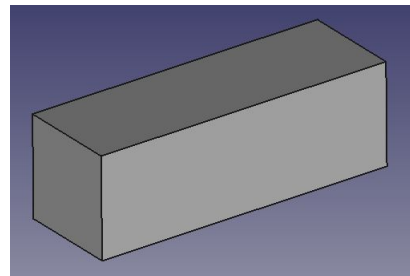
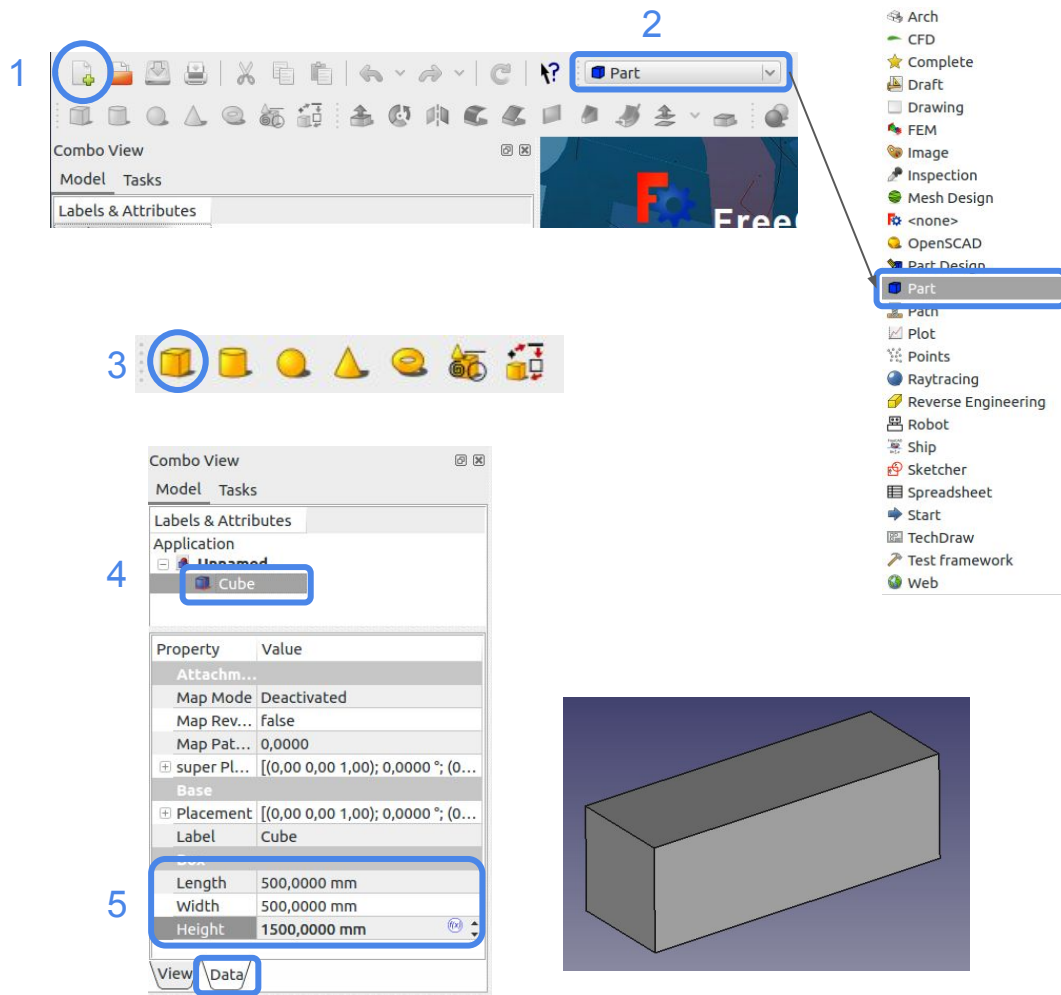
Qingfeng Xia (2015)

Part Design

Create Cube

- As part of this tutorial, we are going to create 3 cubes, and combine them together into one larger shape.
- To create the first cube, activate the “Part” workbench.
- Click on the predefined primitive “Cube” icon.
- To change the cube’s length, width and height:
 - Highlight the “Cube” object
 - Within the “Data” tab, change the properties

Length: 500mm, Width: 500mm, Height: 1500mm



Add 2 More Cubes

- Create 2 More Cubes:

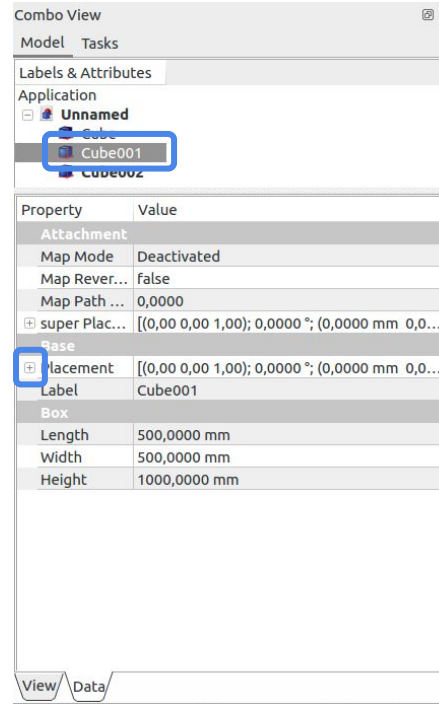
Cube001: Length: 500mm, Width: 500mm, Height: 1000mm

Cube002: Length: 500mm, Width: 500mm, Height: 1500mm

- Move the 2 cubes
 - Click on “Placement” in the data tab.
 - Edit the z-”Position” property to move the cubes.

Cube001: z=1500mm

Cube002: z=2500mm

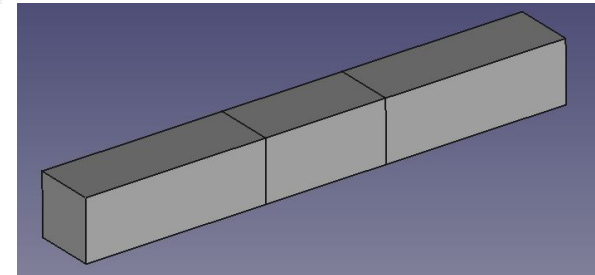


Cube001

Placement	[(0,00 0,00 1,00); 0,0000 °; (0,0000 mm 0,0...
Angle	0,0000 °
Axis	[0,00 0,00 1,00]
Position	[0,0000 mm 0,0000 mm 1500,0000 mm]
x	0,0000 mm
y	0,0000 mm
z	1500,0000 mm

Cube002

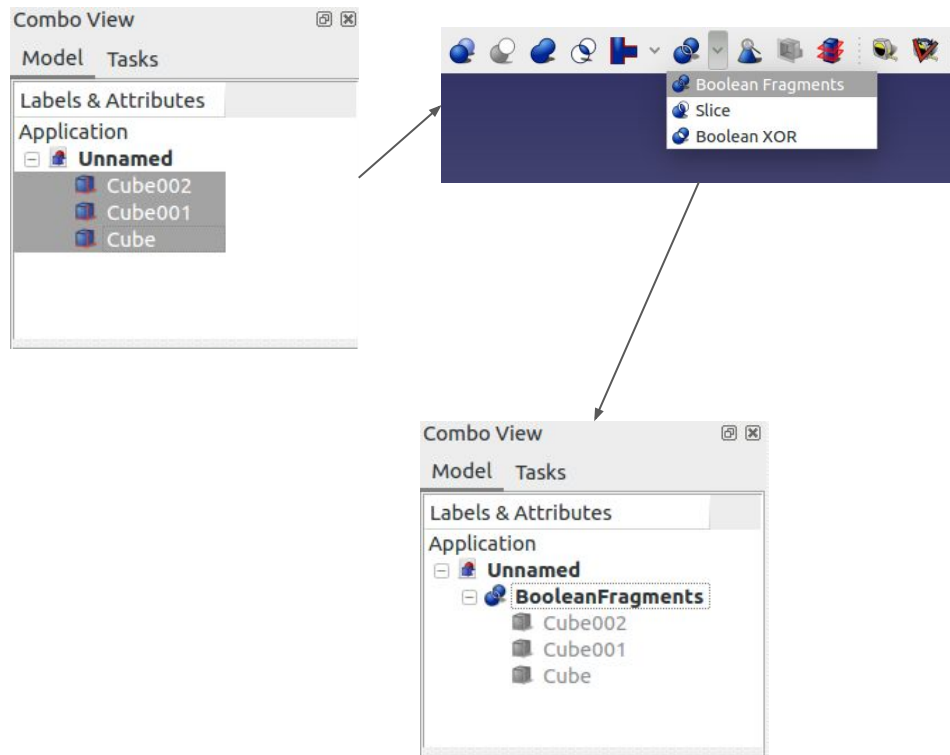
Placement	[(0,00 0,00 1,00); 0,0000 °; (0,0000 mm 0,0...
Angle	0,0000 °
Axis	[0,00 0,00 1,00]
Position	[0,0000 mm 0,0000 mm 2500,0000 mm]
x	0,0000 mm
y	0,0000 mm
z	2500,0000 mm



Boolean Fragments Operation

- The blocks have to be joined using Boolean Fragments.
- *Boolean Fragments* is currently the only known way to guarantee that the multi-region mesh has matching nodes along the interfaces joining multiple regions. None of the other Boolean operators guarantees this.
- Select all 3 cubes, then click on the “Boolean Fragments” icon.

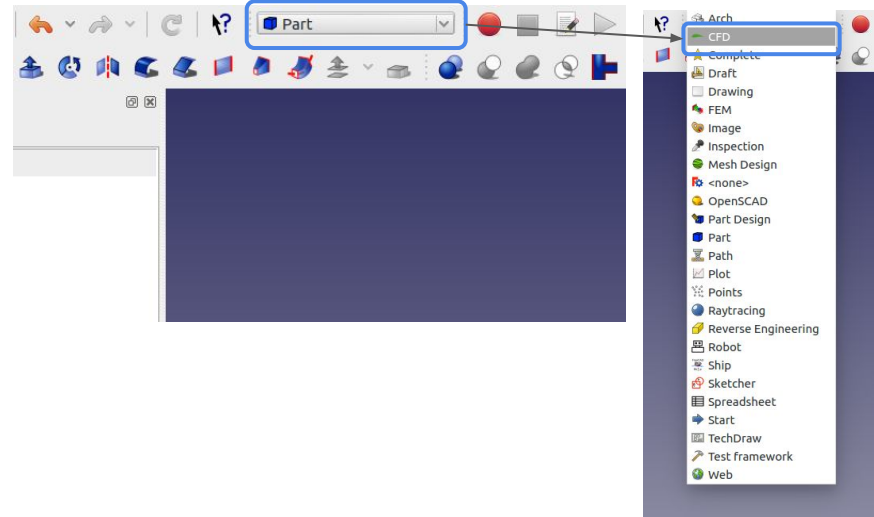
NOTE: Boolean Fragments requires OCC 6.9.0+



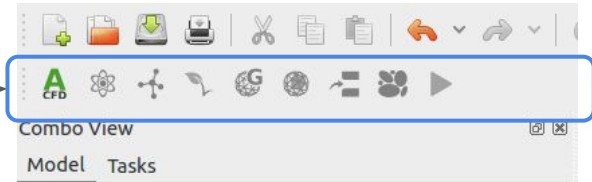
Multi-region mesh generation

Activate Cfd Workbench

- To activate the Cfd Workbench, click on the dropdown menu in the taskbar, and select “CFD”
- Once activated, the Cfd task bar should appear.

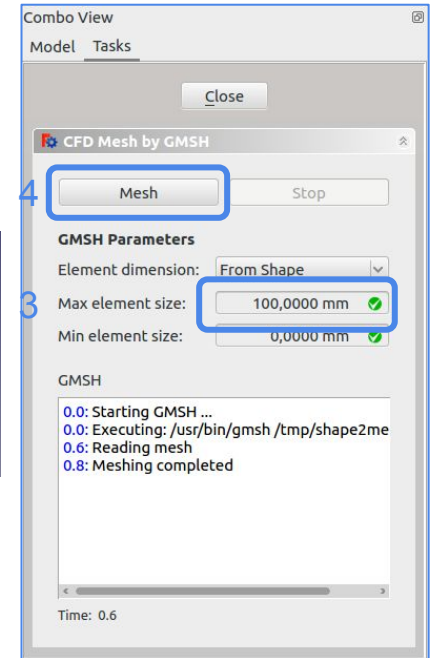
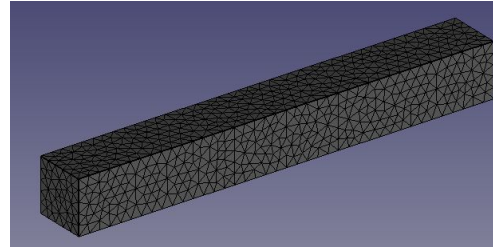
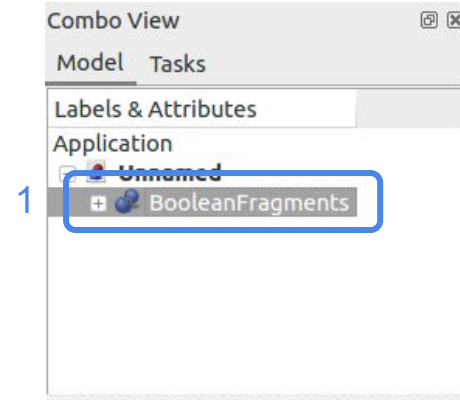


Cfd Task bar: →



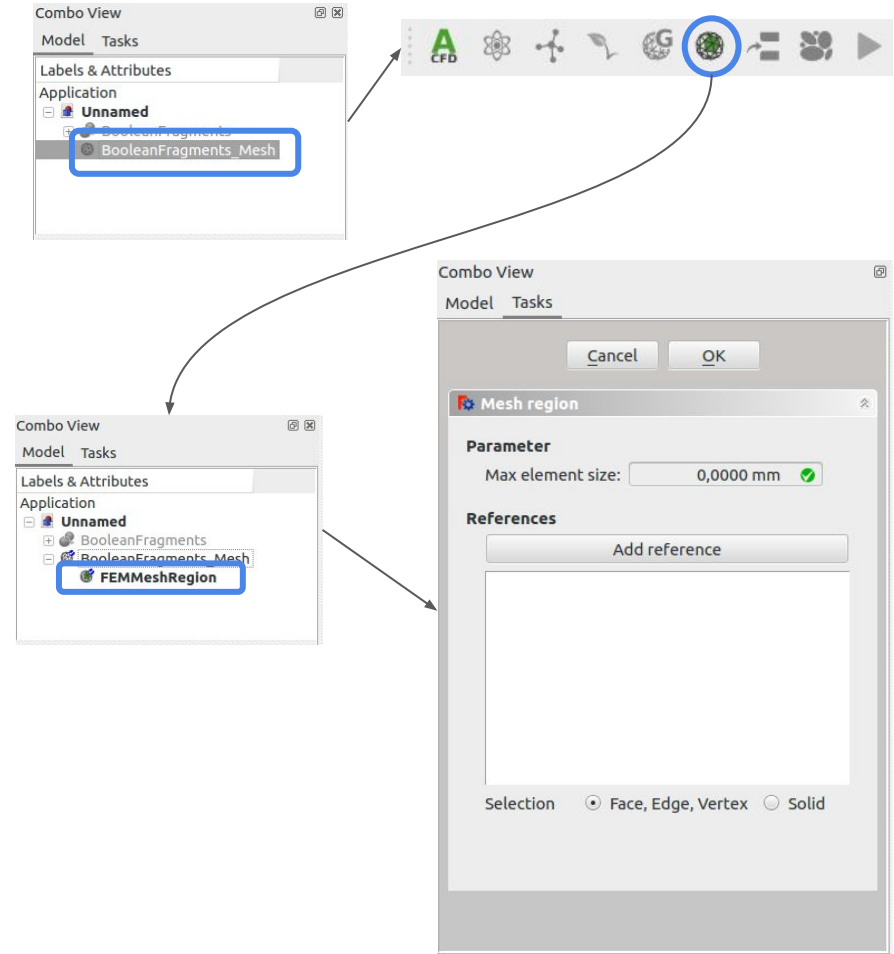
Creating the Initial Mesh

- Let's create a preliminary mesh.
- Highlight the 3D Boolean Fragments object.
- Click on the GMSH icon.
- Set a maximum element characteristic length of 100mm and mesh.
- You should now have a uniformly spaced tetrahedral mesh, where the nodes falling along the faces between the 3 cubes are now guaranteed to be matching.



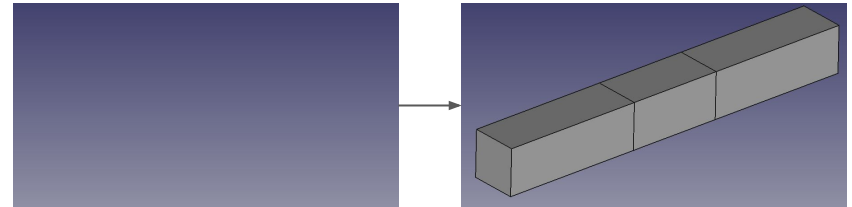
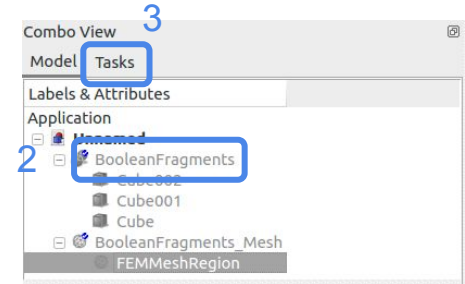
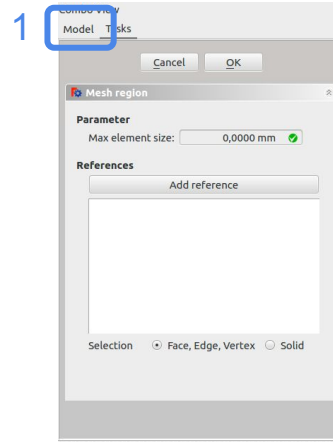
Refining Selected Regions

- Highlight the mesh object, which will activate the mesh region icon
- Click on the mesh region icon (or select “Mesh Region” from the CFD drop down menu).
- A “FEMMeshRegion” object will now have been created (if not visible, expand the “BooleanFragments_Mesh” object by clicking on the “+”).
- Double click “FEMMeshRegion” which will open the “Mesh region” task panel.
- In this task panel, we can now select shapes in the form edges, faces or solids where a different characteristic length can then be prescribed.



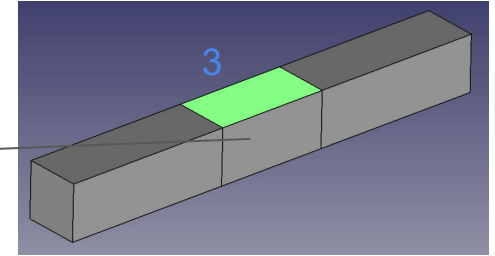
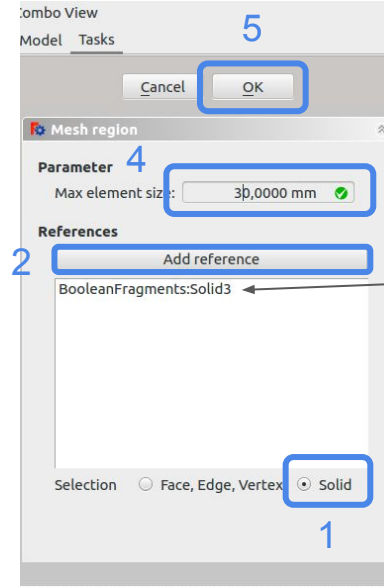
Making selected objects visible

- We wish to now select a specific region for mesh refinement.
- It is possible to only make some of the parts visible, to assist in selecting specific regions, or faces to refine.
- To make specific shapes visible or invisible, click on the “Model” tab.
- Objects can be made visible by clicking on them in the list of objects, and pressing “spacebar”, or right-clicking and selecting “Show Selection” or “Hide Selection”.
- Make the *BooleanFragments* object visible.
- With the 3 cubes now visible, return to the “Mesh Region” editor by clicking on the “Tasks” tab.



Include refined mesh region

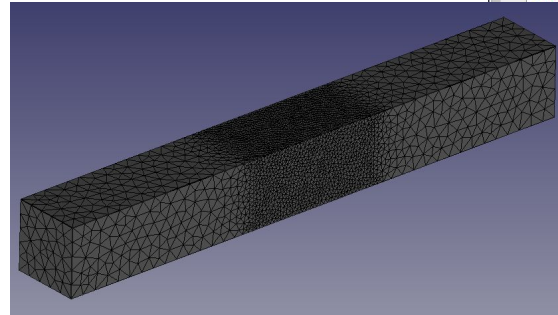
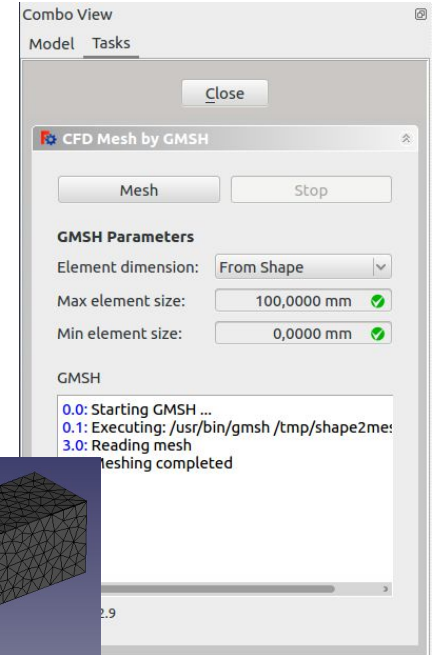
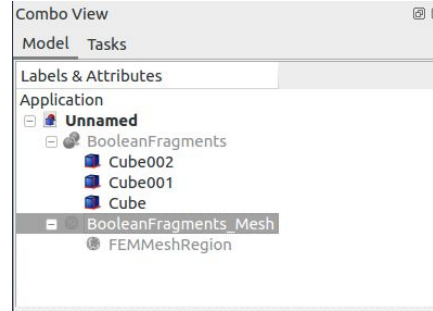
- We wish to refine all the elements within the central cube's region.
- We therefore, first set to Solid selection (other options include a vertex, edge or face).
- Click “Add reference” button, then click on the central cube.
- Set the maximum element size to 30mm.
- Click “OK”.



Re-compute the new mesh

- In the list of objects, double-click on the *parent* (“BooleanFragments_Mesh”) mesh object (or right-click and select “Transform”).
- Click “Mesh” to re-compute the mesh with the new refined region.
- The new mesh should now be locally refined along the centre.

NOTE: The mesh can be refined using, edges, faces or solids. The mesh can be refined with multiple MeshRegions, and does not have to be limited to one. It can also be a mixture of all types of primitives.



Porous-zones

Start a new Cfd analysis

- With the mesh generated, start a Cfd simulation by clicking on the “CFD” icon within the Cfd workbench.
- Because we first generated the mesh, and only then started the Cfd analysis, the mesh will initially fall outside the “CfdAnalysis” container.
- To make the mesh part of the CfdAnalysis, click on the “BooleanFragments_Mesh” and drag and drop it onto the “CfdAnalysis” object. A small “+” symbol should appear next to the mouse.
- If successful, the “BooleanFragments_Mesh” will be an item within the “CfdAnalysis”.

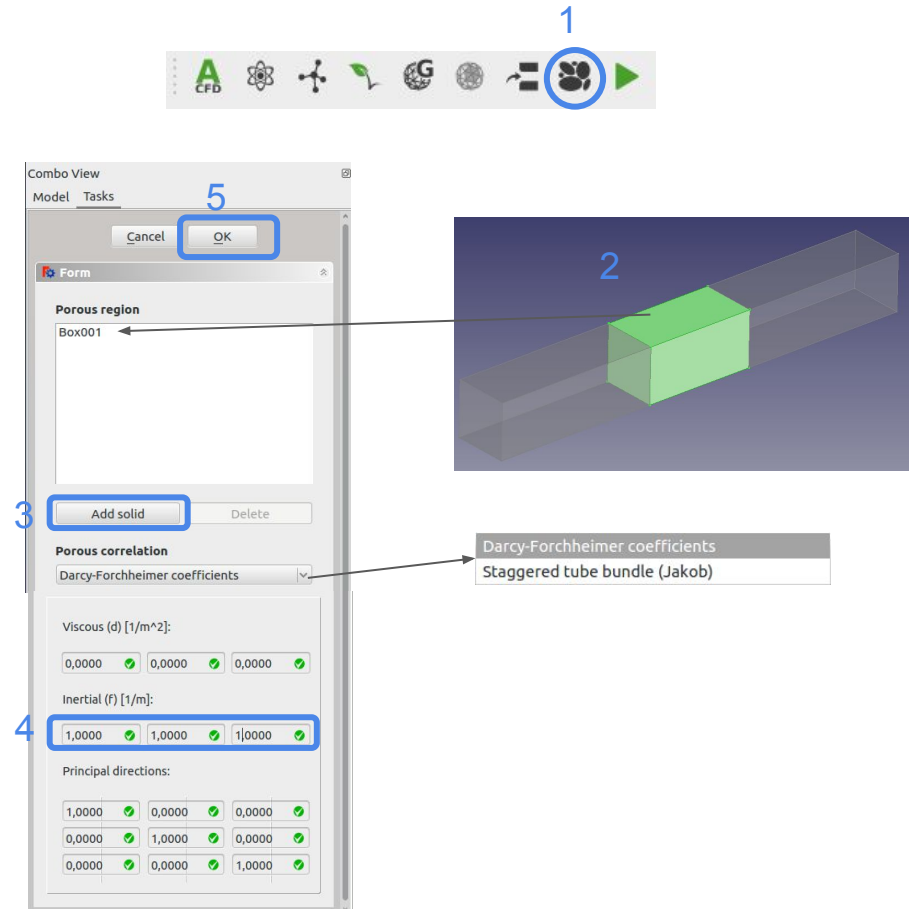
Tip: If the CfdAnalysis object already exists when a mesh is created, the mesh will automatically be part of the CfdAnalysis container.



Adding porous-zone

- Click on the porous-zone icon.
- For the test problem, we wish to make the central cube a porous region.
- Select the central cube and click “Add solid”.
- **NOTE:** Porosity will be added to all the cells contained inside, and intersected by the selected shape (in our case “Box001”).
- There are two types of porosity approximations currently included
 - Darcy-Forcheimer
 - Jakob staggered tube bundle

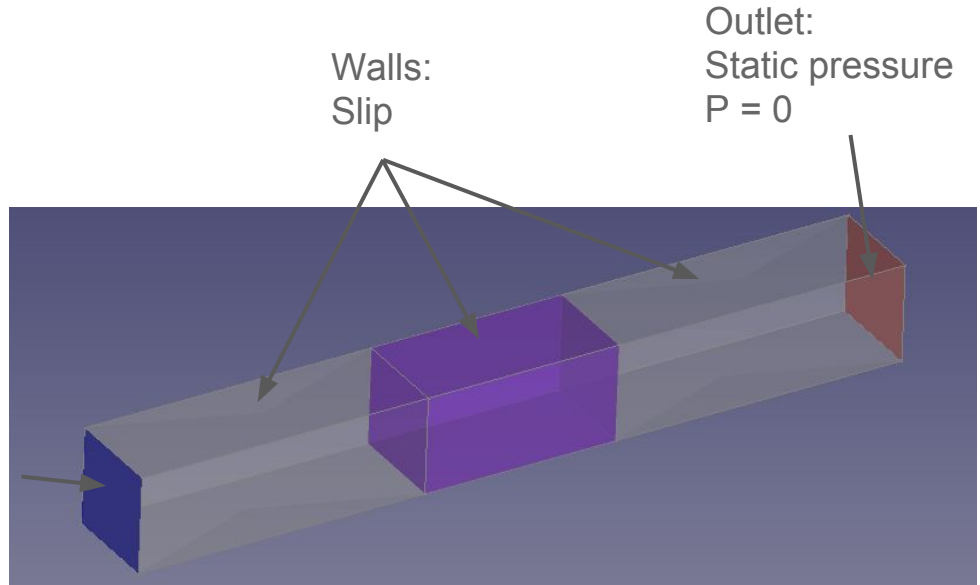
Tip: porous zones are coloured purple once selected.



Adding the remaining boundaries

- Following the instructions provided in Tutorial #1, add boundary conditions, for the inlet, side walls and outlet.
- The remaining settings include:
 - Fluid: air
 - Initialise with potential flow

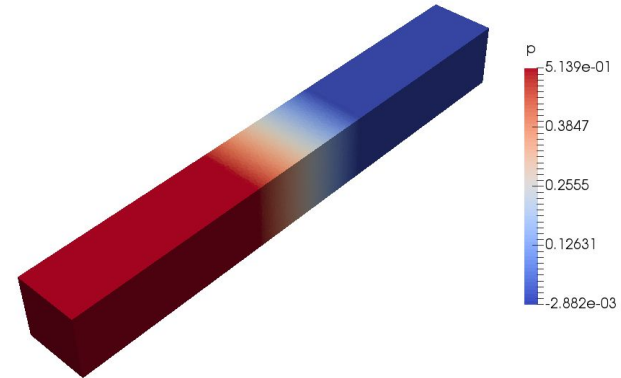
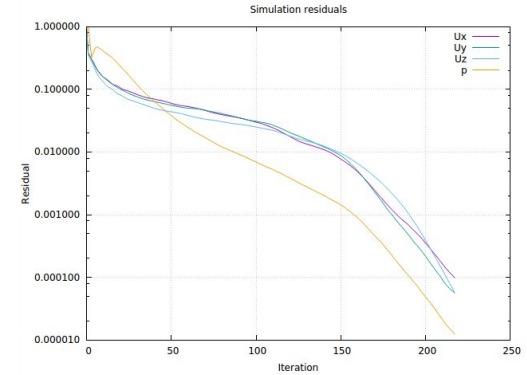
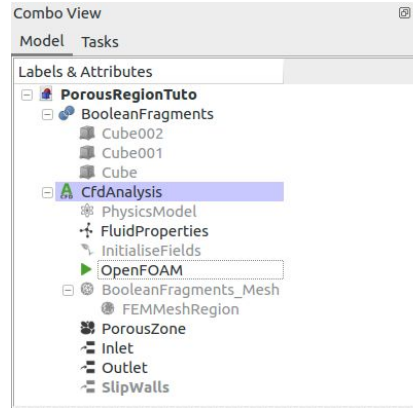
Inlet:
Uniform Velocity
 $U = (0,0,1 \text{ m/s})$



Porous zone results

- With the problem setup we can now write the test case, and run the solver.
- We show here the residual plot and pressure results for the porous zone test problem.
- Notice the large pressure drop across the porous zone.

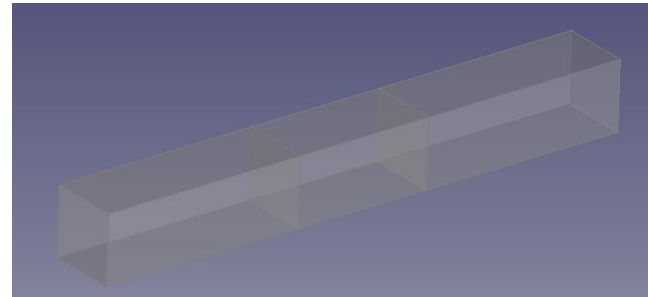
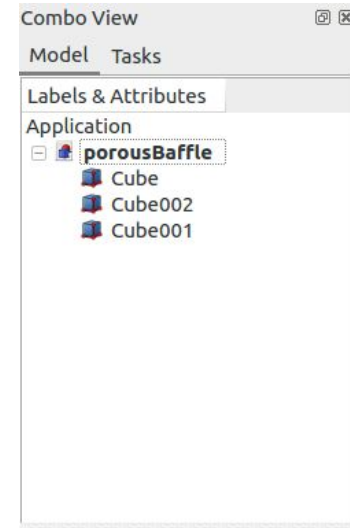
NOTE: The pressure solved for by OpenFOAM is $p = P/\text{density}$.



Porous baffle

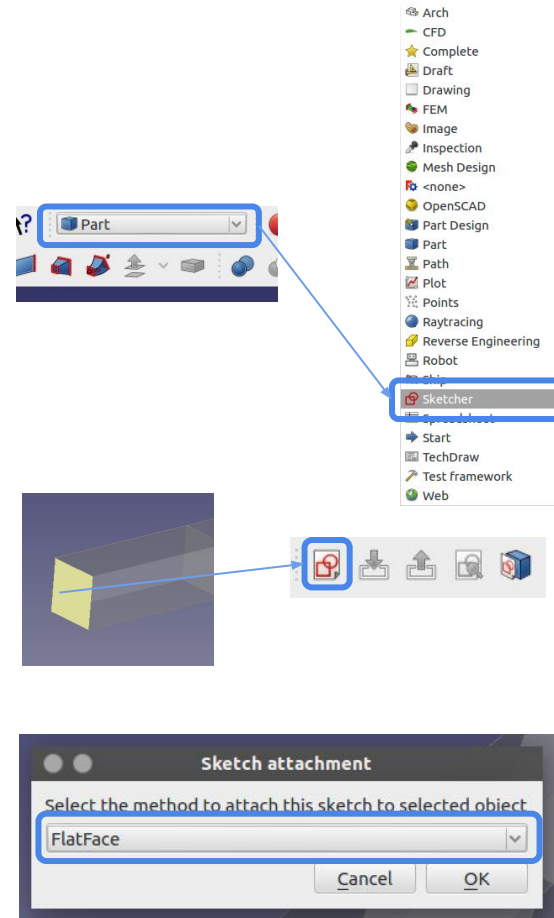
Creating a porous baffle

- A porous baffle is created by adding a corresponding “baffle” boundary condition.
- We therefore need to create a face where the baffle should be and then create a “Boolean Fragments”.
- So let us reset our simulation to just the 3 cubes we created at the start of the tutorial.



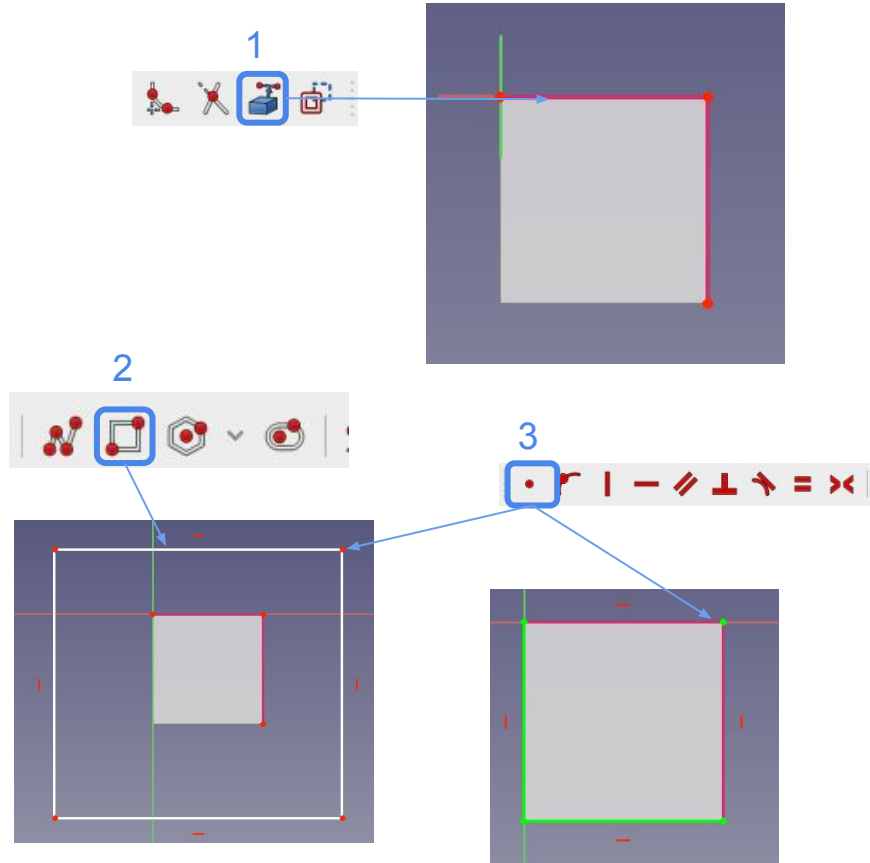
Initialise the sketch

- There are multiple ways of creating faces.
- We are going to do so by creating a sketch of a rectangle, and creating a face from the edges.
- Activate the sketcher workbench.
- Highlight the leftmost face, and click on the “sketcher” icon. This will open a prompt asking where to place the sketch.
- Select “FlatFace”, this is the current face you have highlighted, allowing us to create a sketch directly onto the highlighted face.



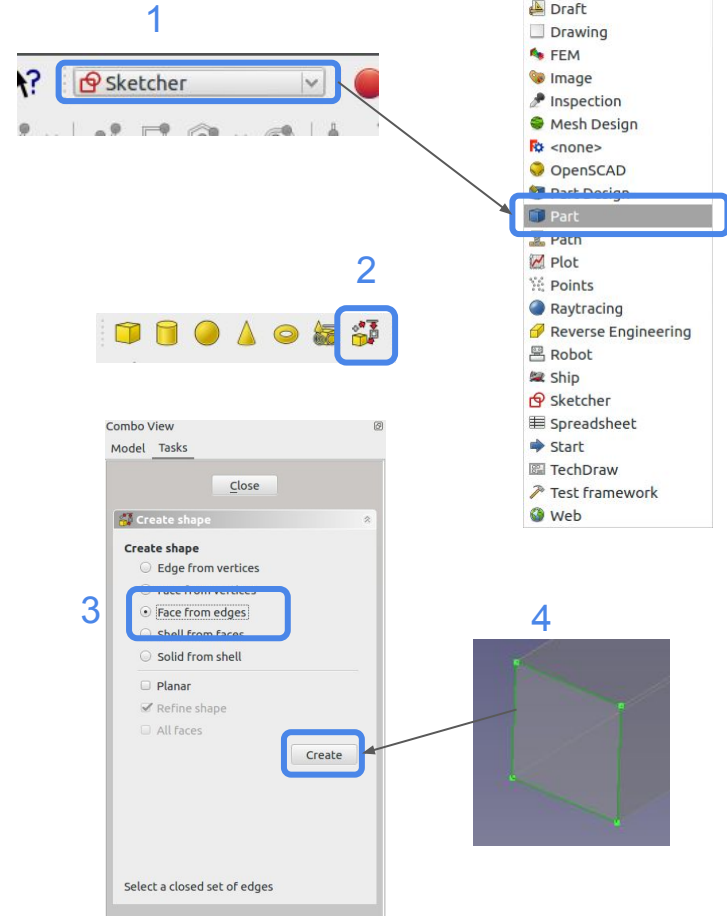
Draw a rectangle

- We start by creating reference lines. These lines link the edges from the already existing Cube object to our newly created sketch.
- To do so, click on the create edge link icon, then select the edges of pre-existing cube face.
- We now draw a rectangle, and make the corners coincident with our reference lines.
- Once all the corners of the rectangle are coincident with the reference lines, there should now be a rectangle of equal size to that of the Cube's face.



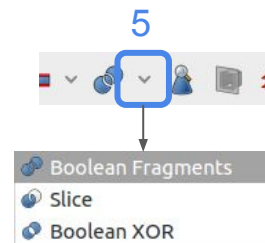
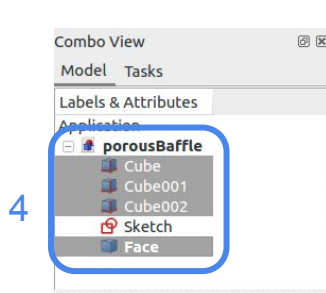
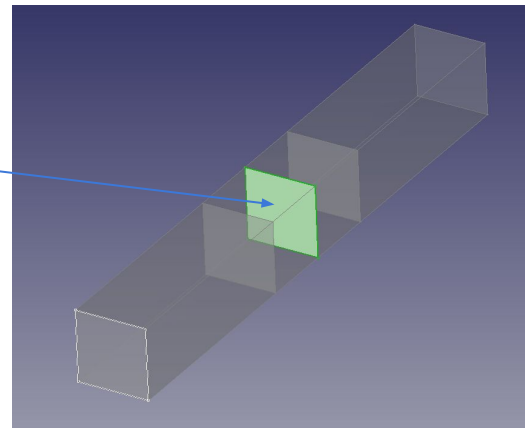
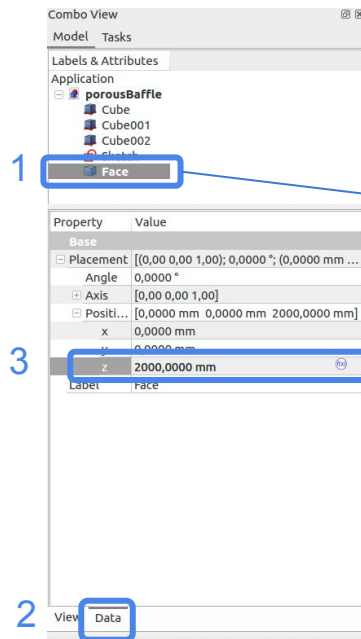
Create face from edges

- In order to change our sketch into a face, activate the “Part” workbench.
- Click on the “Advanced shape utility” icon.
- Select “Face from edges” option.
- Select all the edges of our newly created sketch (NOTE: hold in “Ctrl” to select more than one edge simultaneously).
- Click “Create”.



Create Boolean Fragments

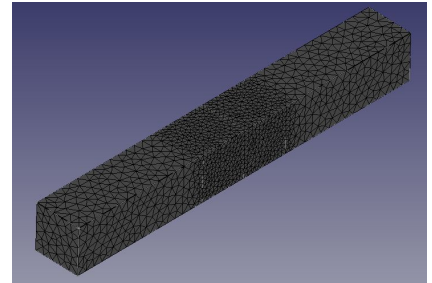
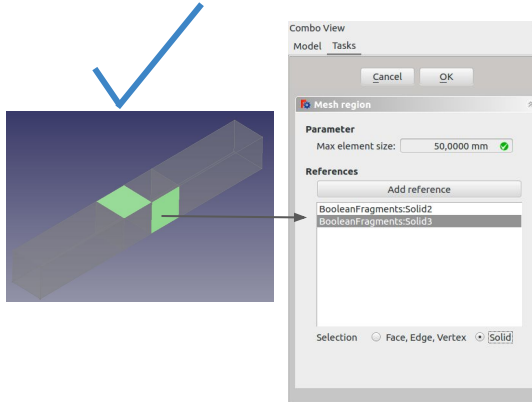
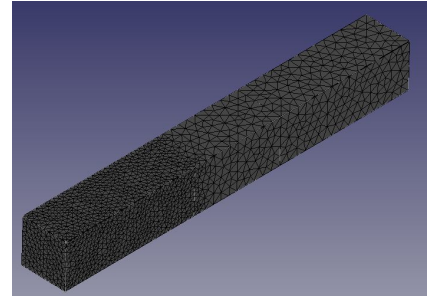
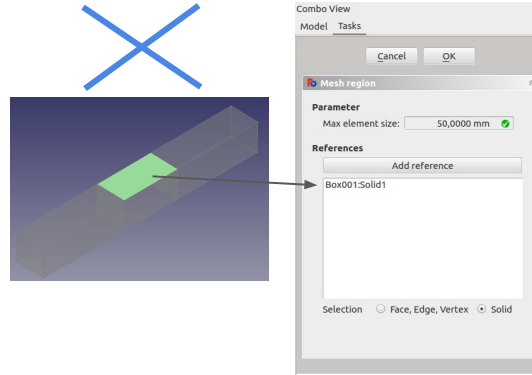
- Move the newly created face to the centre of the 3 cubes. To do so:
 - Highlight the “Face” in the list of objects.
 - Within the “Data” tab, change the z-Position to 2000mm.
- Create a “Boolean Fragments” compound. Select the 3 cubes and the newly created and re-positioned “Face”. Then click on the “Boolean Fragments” icon.



Start CFD analysis and create multi-region mesh

- Start a CFD analysis (selecting “CFD” from the list of workbenches)
- Create a mesh, with characteristic length of 100mm.
- Create a refined region in the centre using the multi-region mesh generation.

NOTE: When creating a multi-region mesh, the user has to specify shapes that exist in the final compound “BooleanFragments” shape. For example, in the current problem, the central cube has been cut into two smaller cubes by our baffle *Face*. To refine the central region, the user must therefore specify the two smaller cubes and not the original large central cube. Consider the example shown in the top-right. The central cube is selected but the leftmost region gets refined. This occurs because the central cube no longer exists within the final shape. Selecting the two smaller cubes directly on the “BooleanFragments” shape results in the expected behaviour.

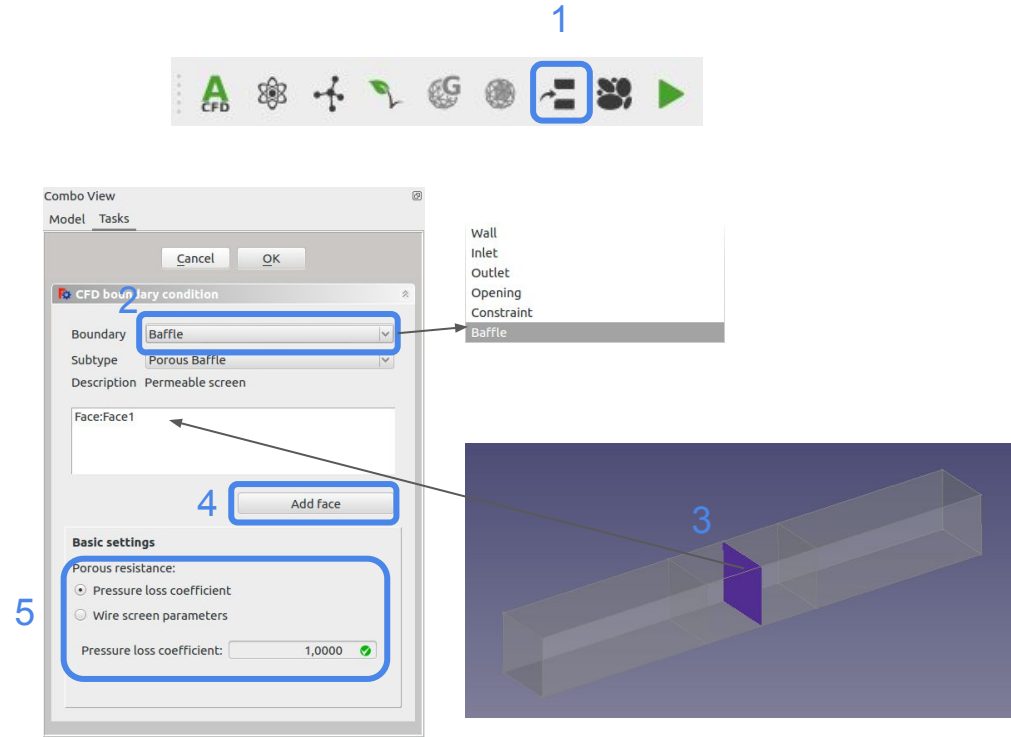


Add porous baffle

- A porous baffle is added by adding the “Baffle” boundary condition.
- Click on the boundary condition icon.
- Select “Baffle” from the Boundary Type.
- Click on the “Face” and click “Add face”
- Specify either the “Pressure loss parameters” or “Wire screen parameters”.

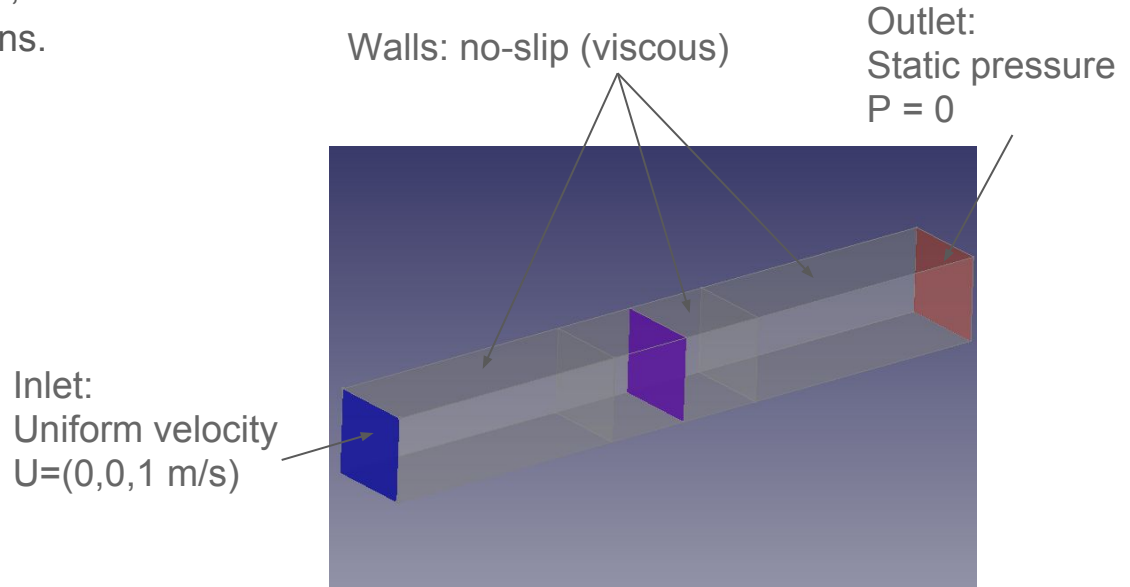
TIP: Recall, that various objects can be made visible or hidden in the “Model” tab by highlighting the objects and pressing spacebar (or right-clicking and selecting Show/Hide). This may be necessary to select the “Face”.

TIP: The baffle Face can alternatively be added by highlighting the “Face” object in the list of objects under the “Model” tab, and then clicking “Add face” within the still open CFD boundary condition “Tasks” tab.



Adding the remaining boundaries

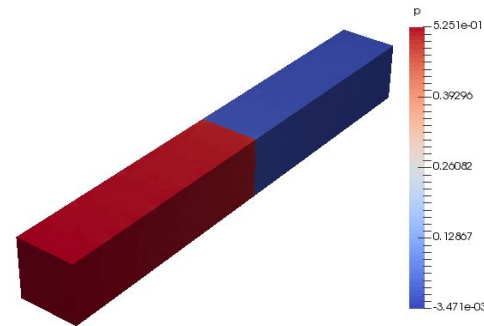
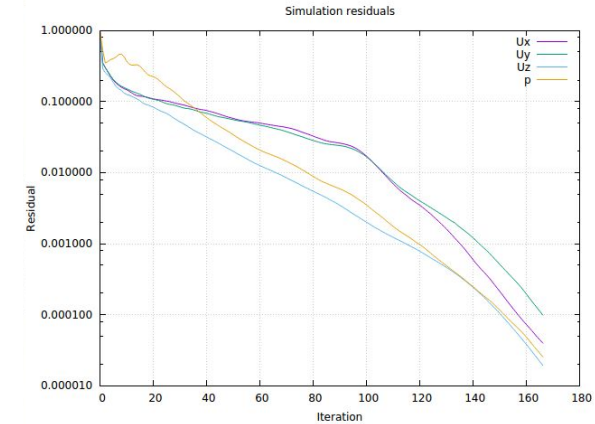
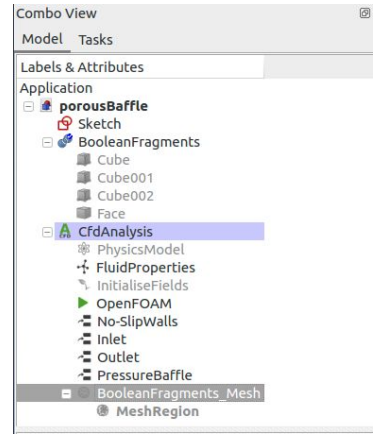
- Following the instructions provided in Tutorial #1, add the remaining inlet, side walls and outlet boundary conditions.
- The remaining settings include:
 - Fluid: air
 - Initialise with potential flow



Porous baffle results

- With the problem setup we can now write the test case, and run the solver.
- We show here the residual plot and pressure results for the baffle test problem
- Notice the large distinct pressure drop across the porous baffle.

NOTE: The pressure solved for by OpenFOAM is the density normalised pressure:
 $p = P/\text{density}$.



The End